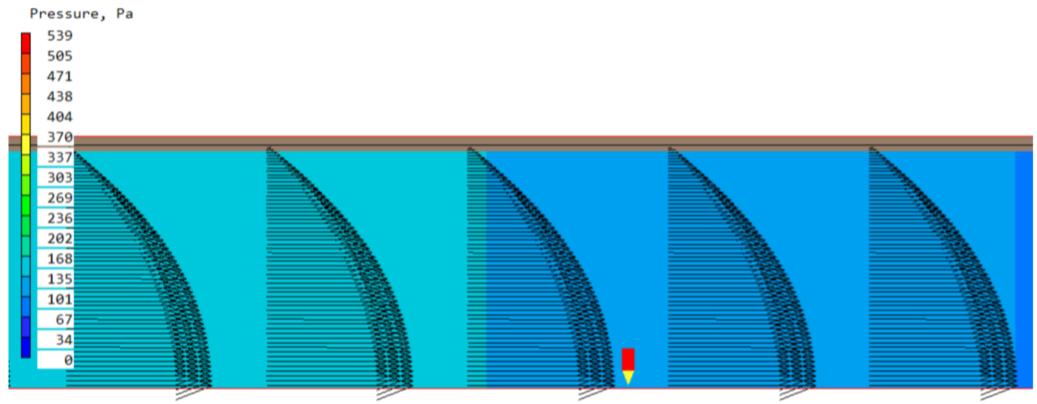


CFD test case 1

Laminar oil transport in a pipeline

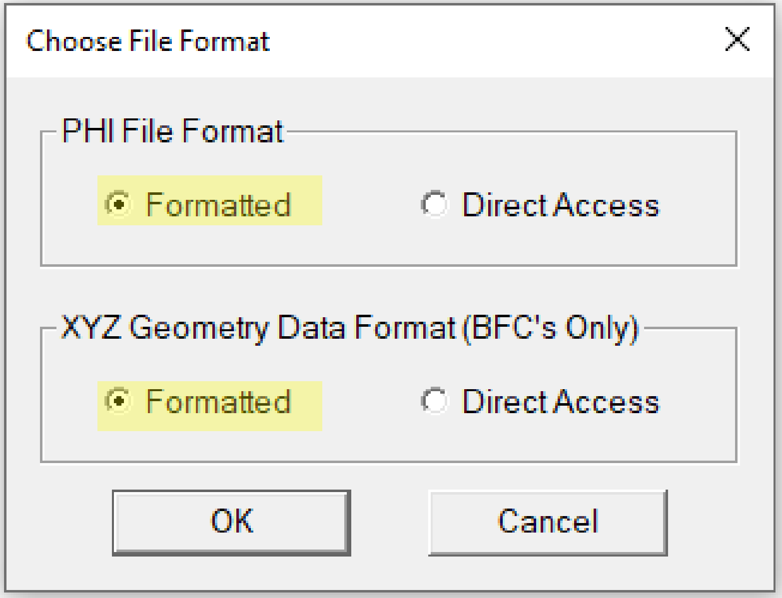
Suggestions for setting up the case

Ing. Luca Nicola Quaroni
Ing. Jorge Soto
Ing. Federico Lanteri
Prof. Gianandrea Vittorio Messa

POLITECNICO
MILANO 1863

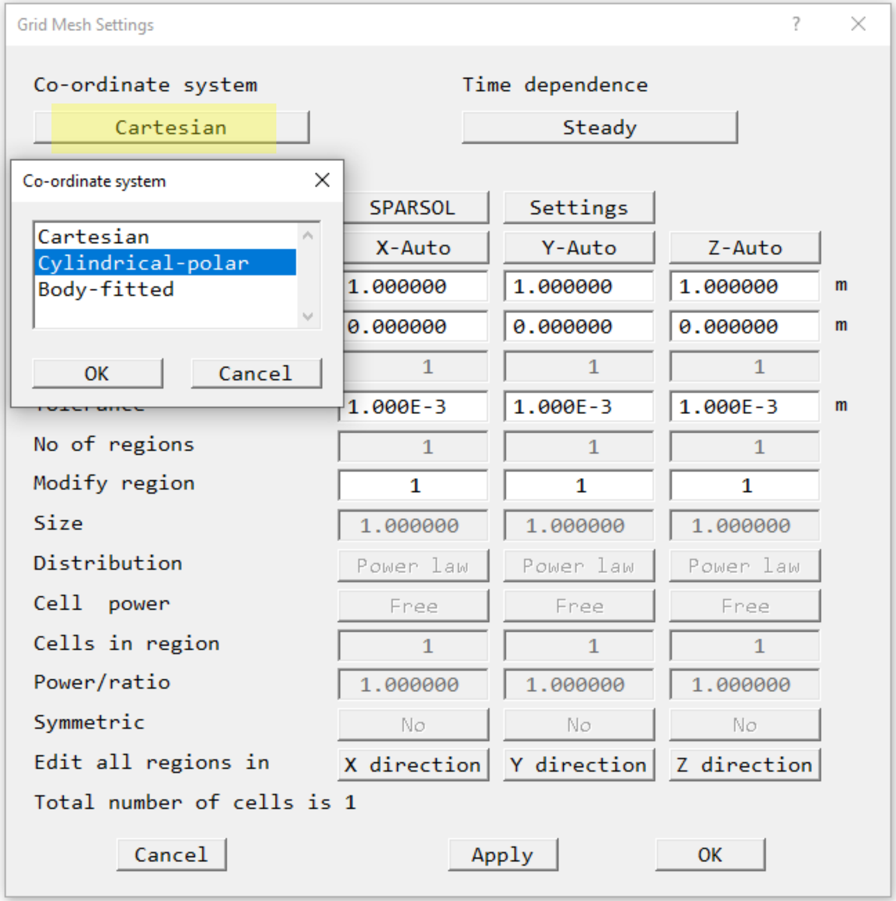
Data formatting

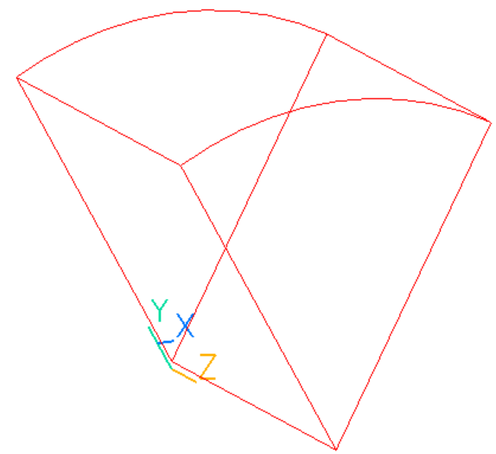
Go to *Options* → *File Format.* Set both queries to *Formatted* for use of post-processing Matlab codes.



Geometry
Coordinate system

Go to *Settings* → *Domain Attributes* → *Geometry* → click on «*Cartesian*» and select *Cylindrical-polar*

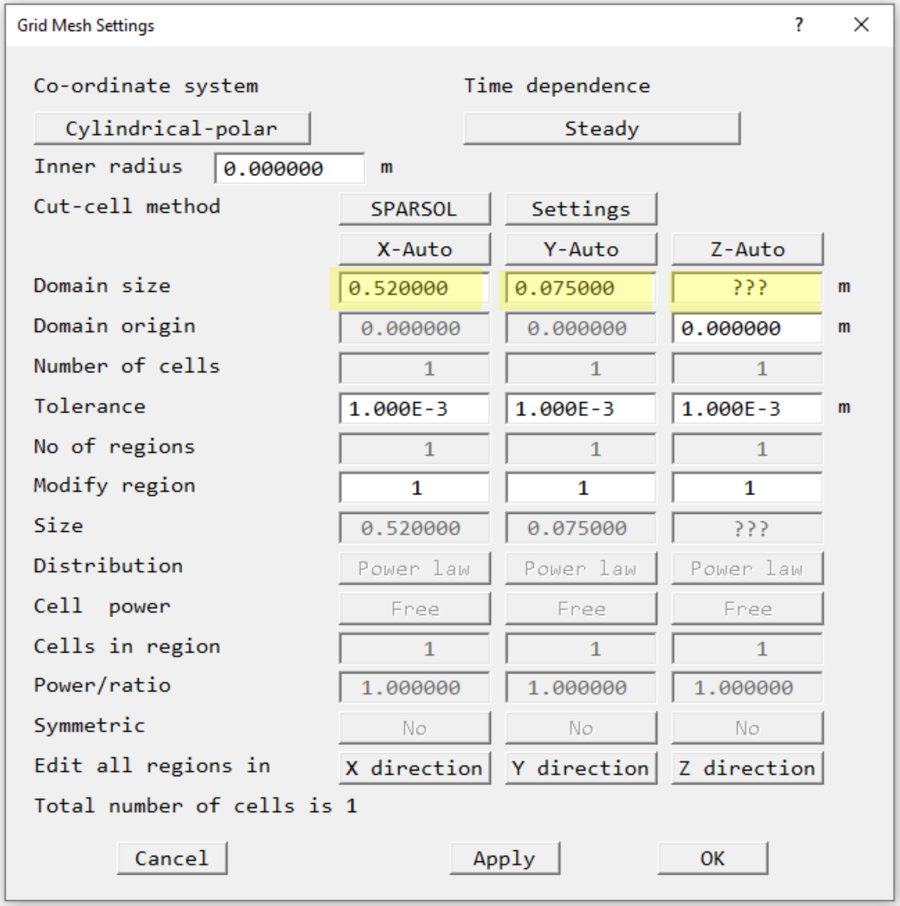




$$X = \theta, Y = r, Z = z$$

Geometry
Domain size

In the same panel, set the dimensions of the numerical domain (X is in rad)

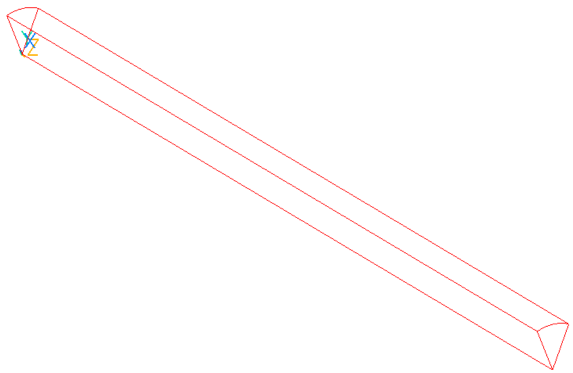

Grid Mesh Settings

Co-ordinate system: Cylindrical-polar
Time dependence: Steady
Inner radius: 0.000000 m
Cut-cell method: SPARSOL | Settings

	X-Auto	Y-Auto	Z-Auto	
Domain size	0.520000	0.075000	???	m
Domain origin	0.000000	0.000000	0.000000	m
Number of cells	1	1	1	
Tolerance	1.000E-3	1.000E-3	1.000E-3	m
No of regions	1	1	1	
Modify region	1	1	1	
Size	0.520000	0.075000	???	
Distribution	Power law	Power law	Power law	
Cell power	Free	Free	Free	
Cells in region	1	1	1	
Power/ratio	1.000000	1.000000	1.000000	
Symmetric	No	No	No	
Edit all regions in	X direction	Y direction	Z direction	

Total number of cells is 1

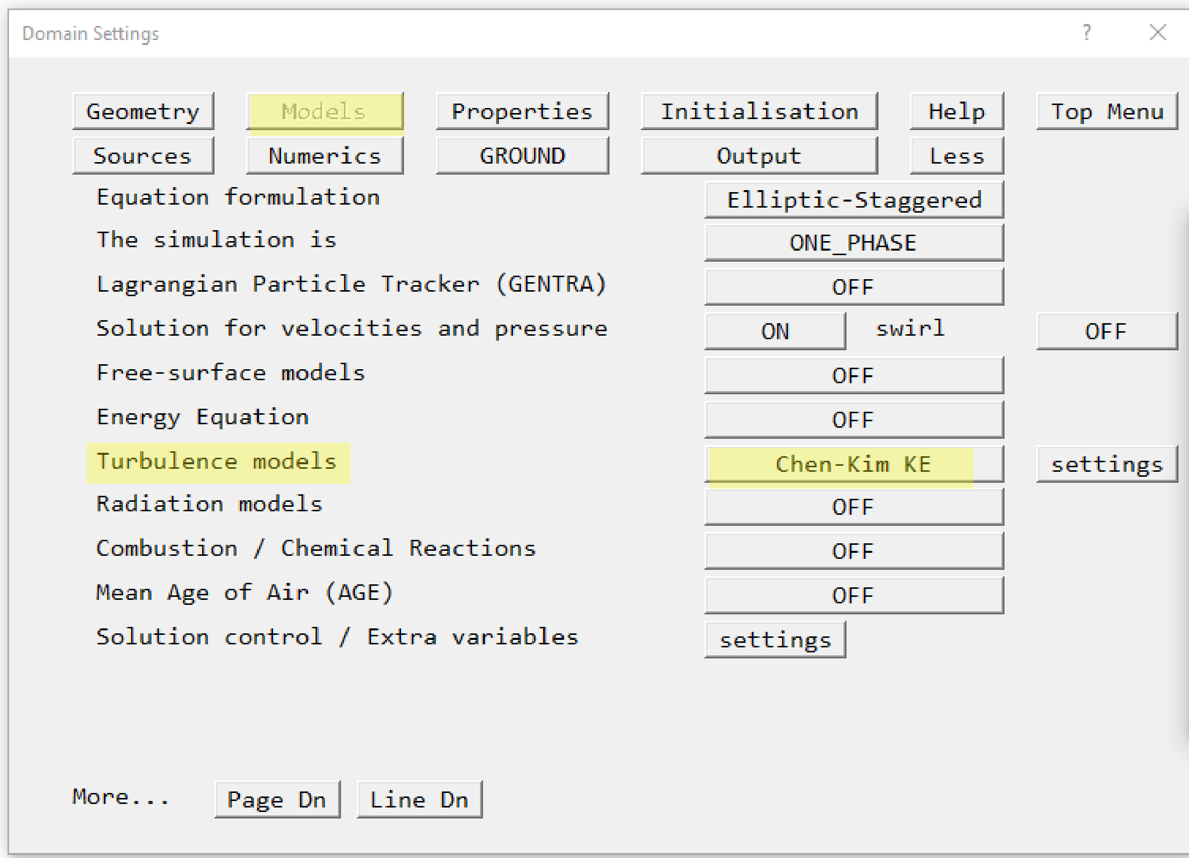
Cancel | Apply | OK

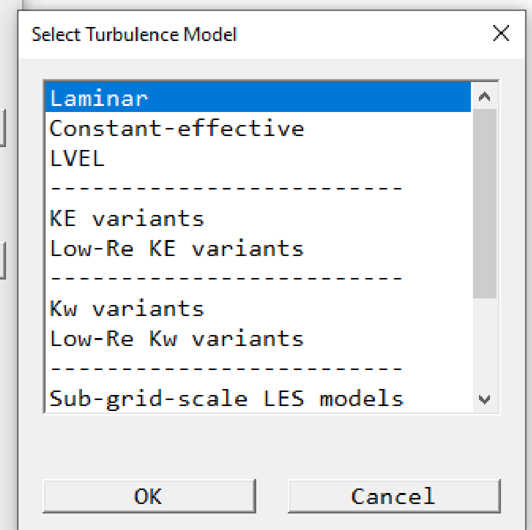


N.B. Axi-symmetry is automatically imposed by PHOENICS

Models
Turbulence model

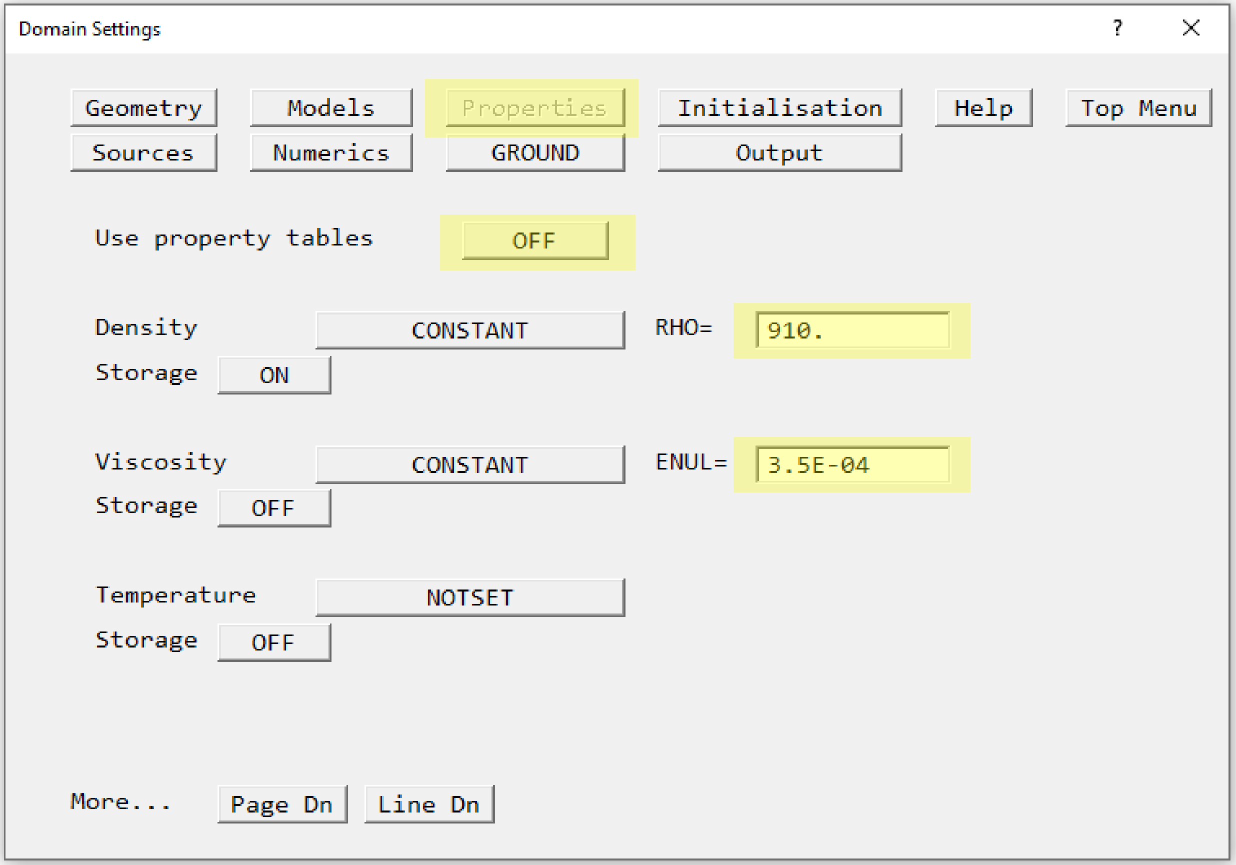
Go to *Settings* → *Models* → *Turbulence models*. Click on the default model and select *Laminar.*





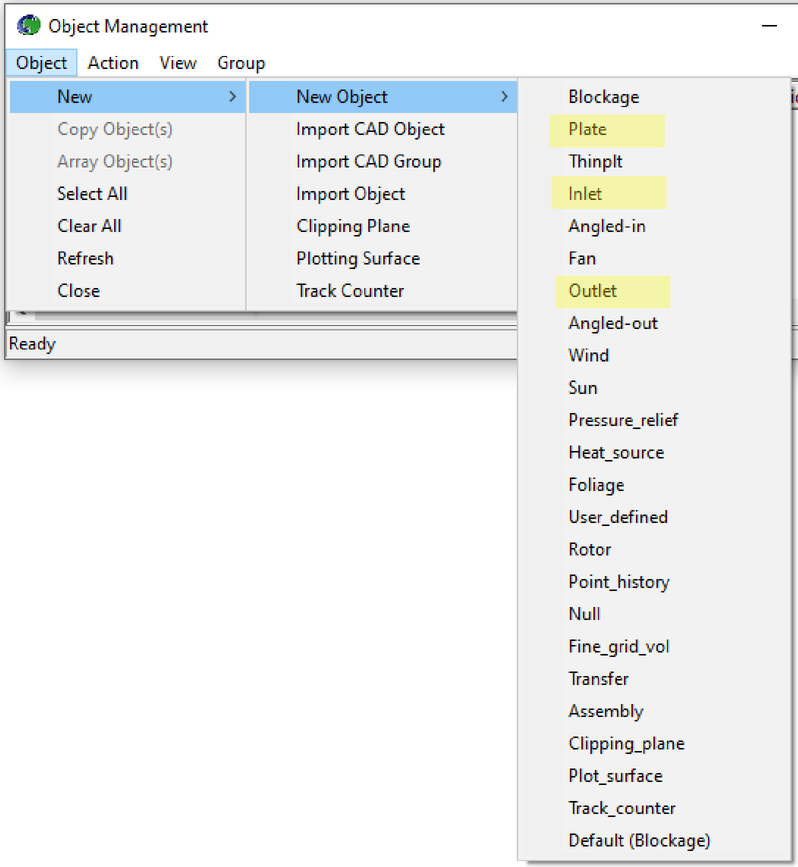
Fluid properties

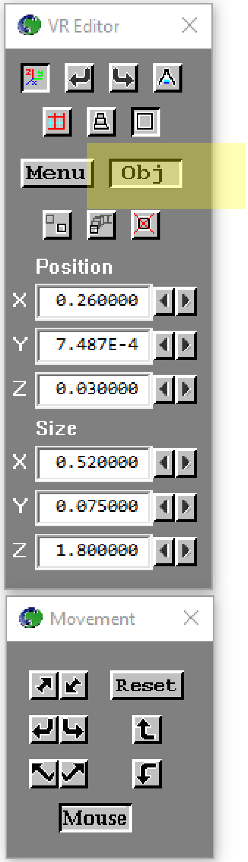
Go to *Settings* → *Properties* → Switch OFF *Use property tables*. Set the density to 910.0 kg/m³ and the kinematic viscosity to 0.00035 m²/s.



Boundary conditions

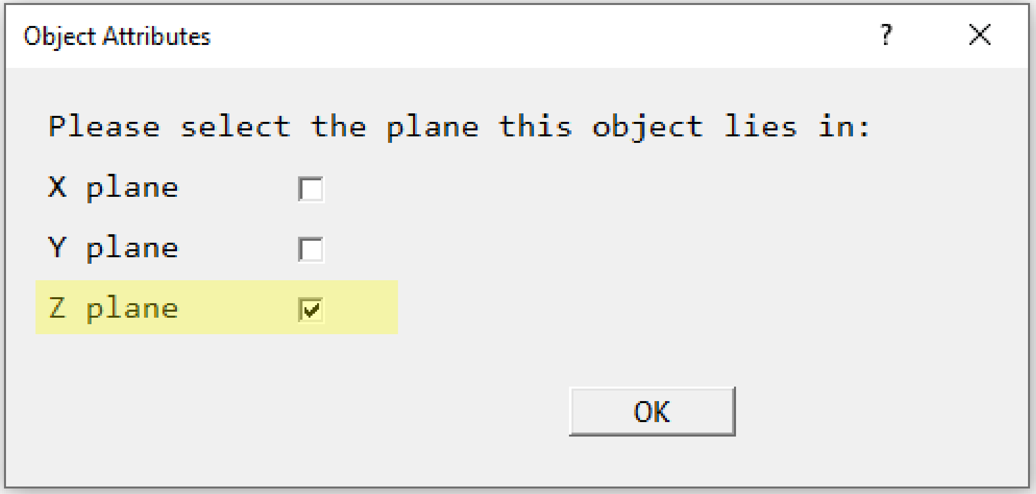
Boundary conditions can be set through *Obj* in VR Editor window. On the *Object Management* window, go to *Object* → *New* → *New Object*; a list will appear.





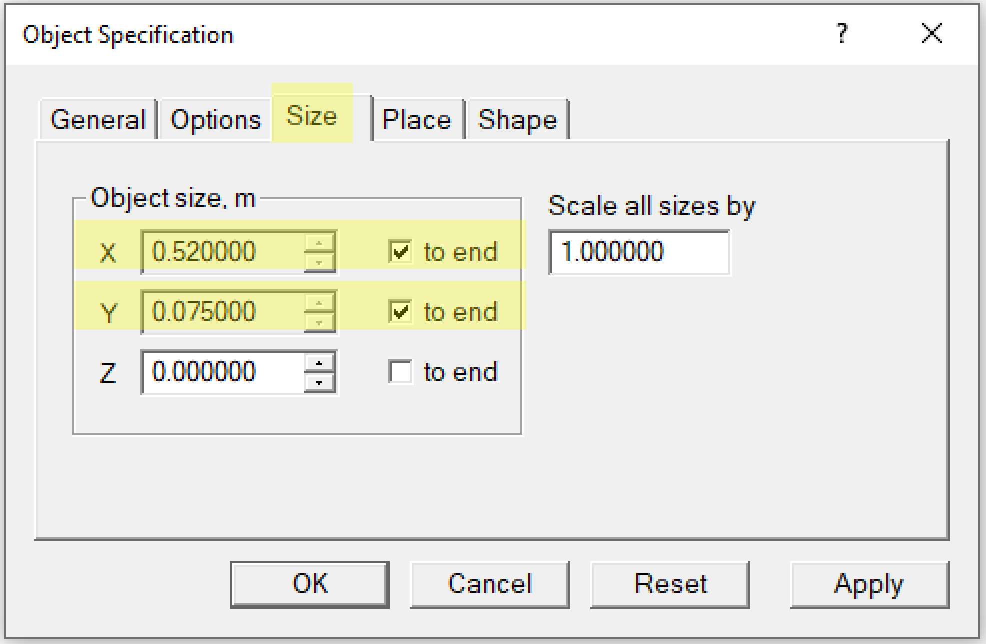
Boundary conditions
Inlet

Select *Inlet* and toggle on the Z plane for its position.



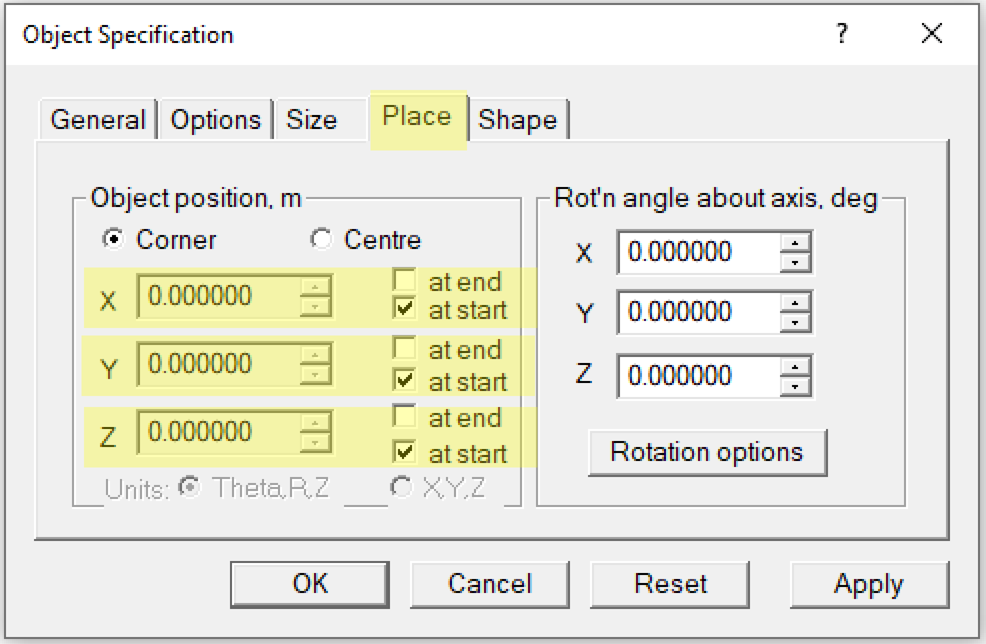
Boundary conditions
Inlet

On the *Object Specifications* window, go to *Size*. Set inlet size *to end* for both *X* and *Y* directions.



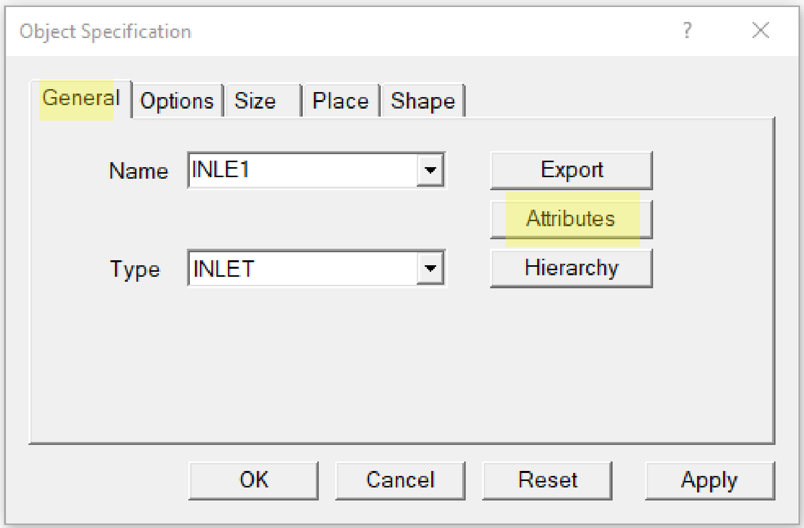
Boundary conditions
Inlet

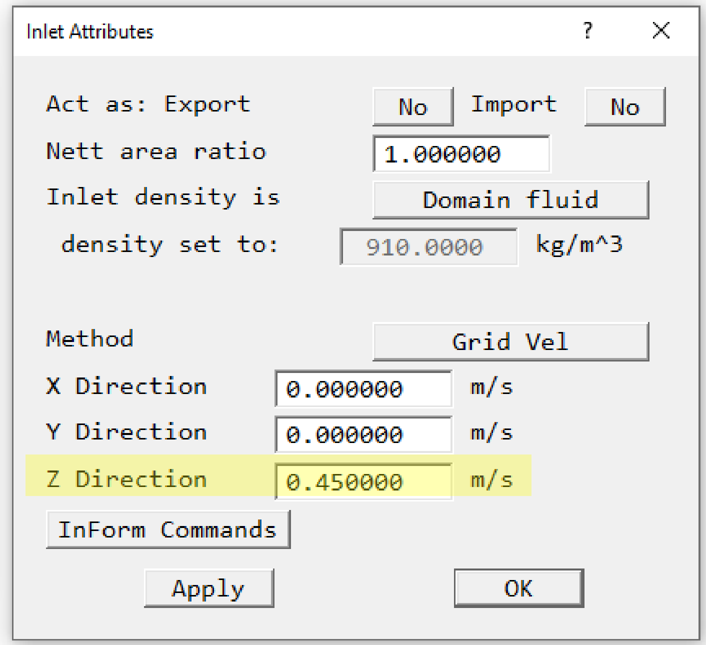
On the same window, go to *place*. Set inlet position *at start* for all dimensions.



Boundary conditions
Inlet

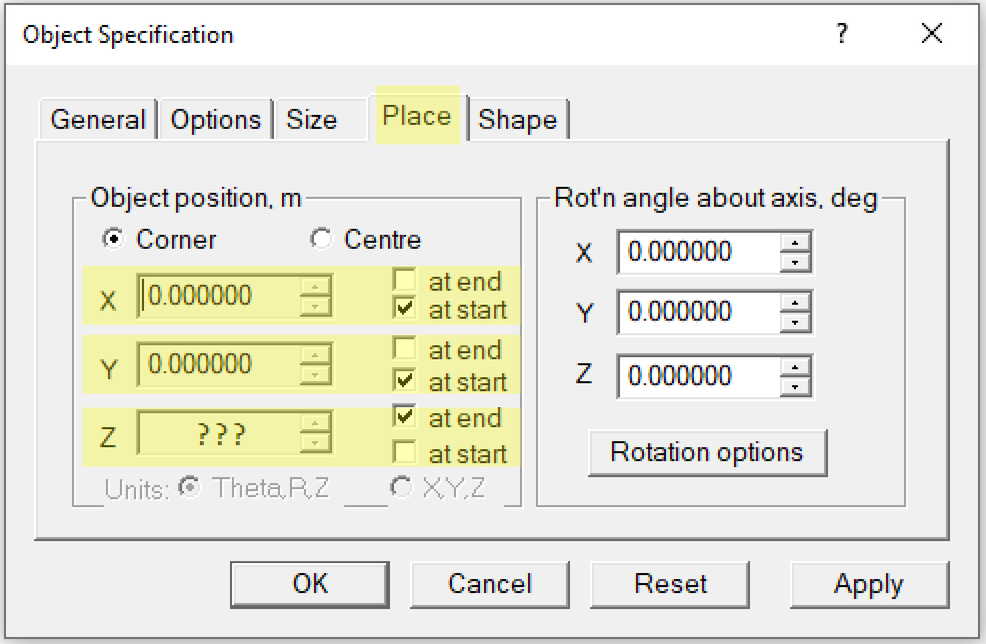
On the same window, go to *General* → *Attributes.* Set the axial velocity to 0.45 m/s.





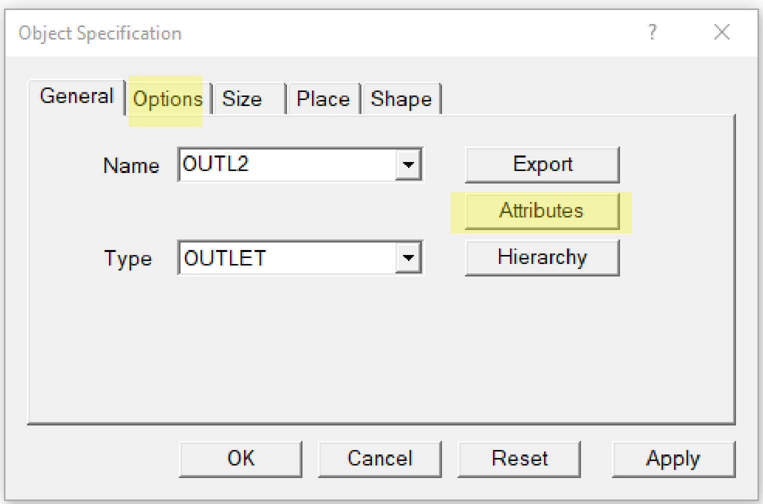
Boundary conditions
Outlet

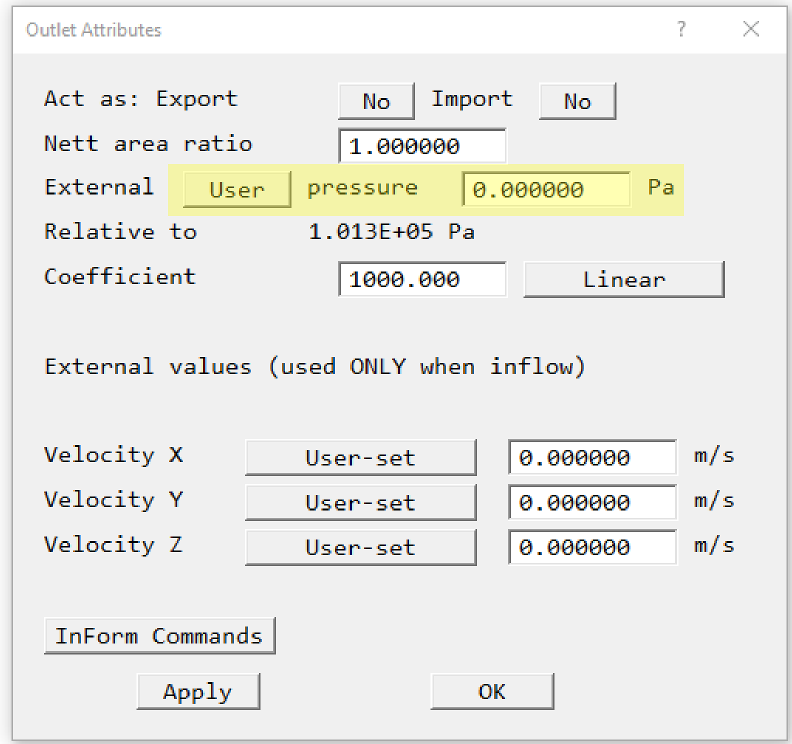
From the *New Object* window, select *Outlet* and repeat the same steps as for the *Inlet* for plane and *size.* Set the position *at start* for both *X* and *Y* directions. Set the position *at end* for the Z-axis:



Boundary conditions
Outlet

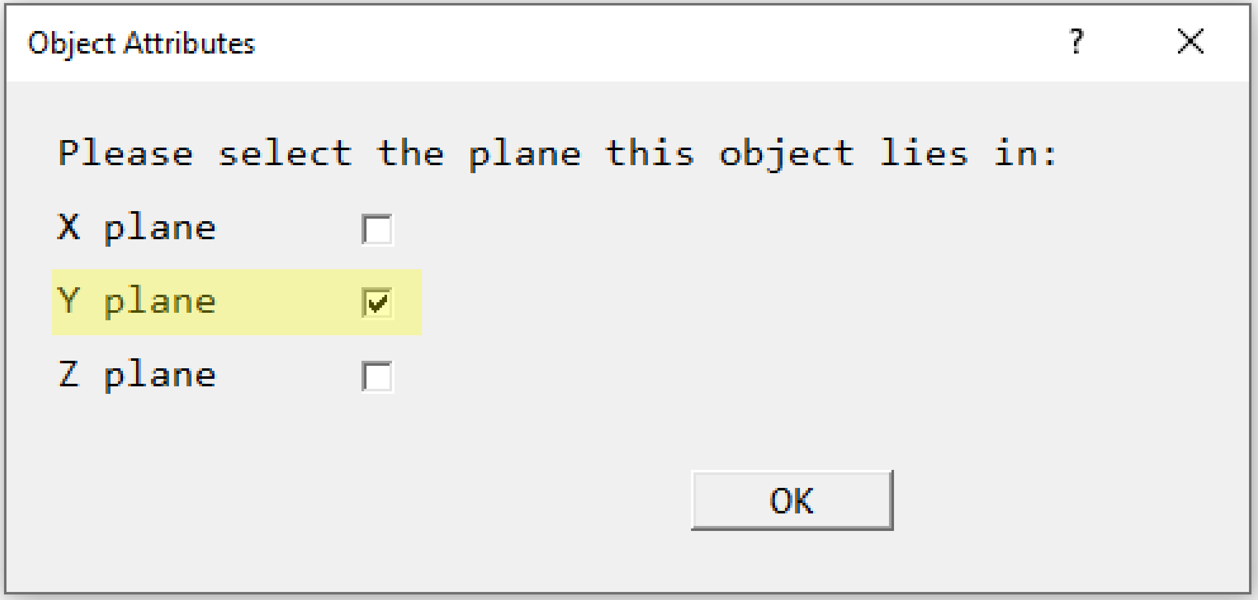
On the *Attributes* window, set the *external pressure* to *user* and impose *0.000000 Pa*.
Suggestion for individual study: how should you change the coefficient to have a null outlet pressure?





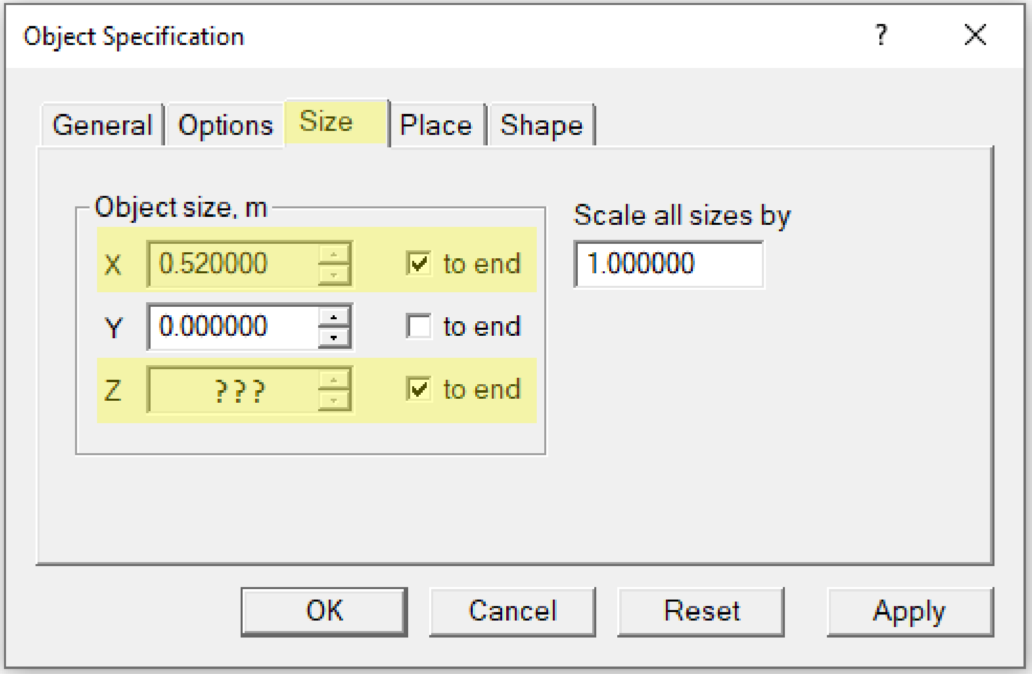
Boundary conditions
Wall

On the *New Object* window, select the *Plate*. The *Plate* object has a zero thickness in the Y plane.



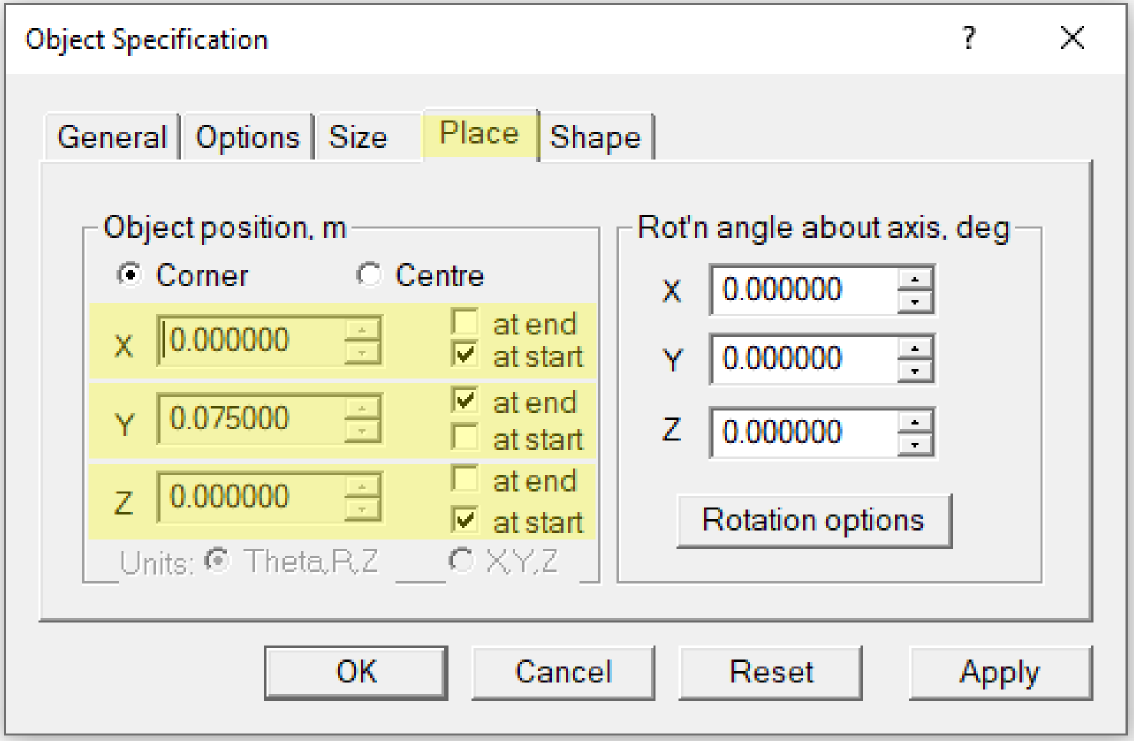
Boundary conditions
Wall

On the *Object Specification* of the plate, set its size to the end for the *X* and *Z* directions.



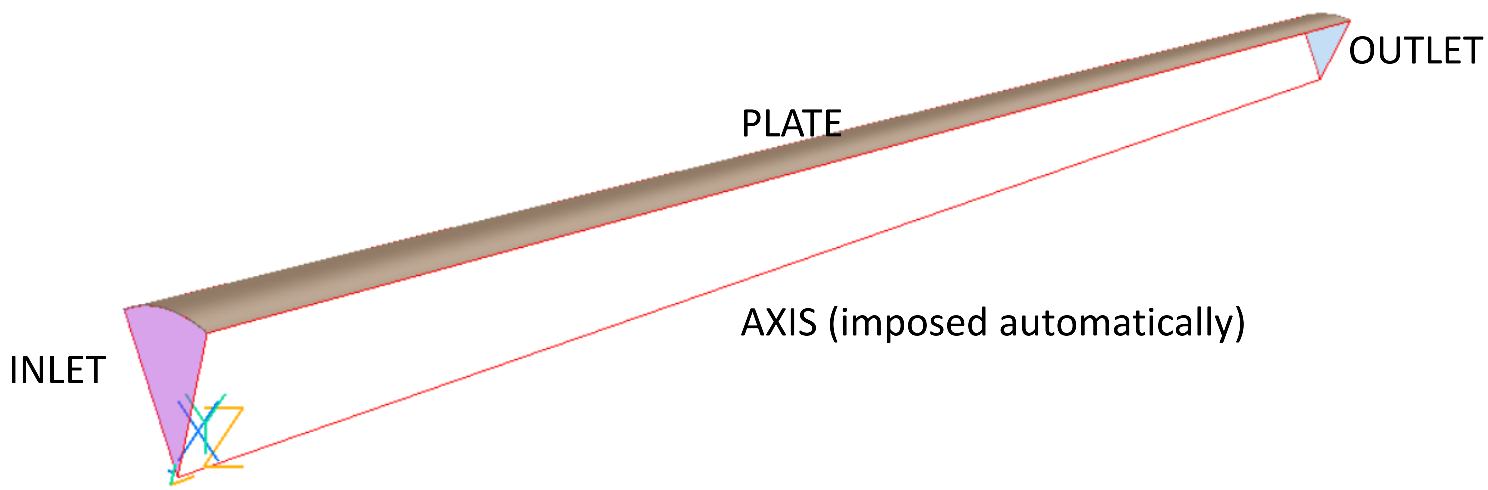
Boundary conditions
Wall

On the *Object Specification* of the plate, set the *Place* at *the end* of the *Y* direction. Set the position *at start* for both *X* and *Z* directions.



Boundary conditions
Wall

The *objects* are visible with different colors.



Mesh
Number of cells

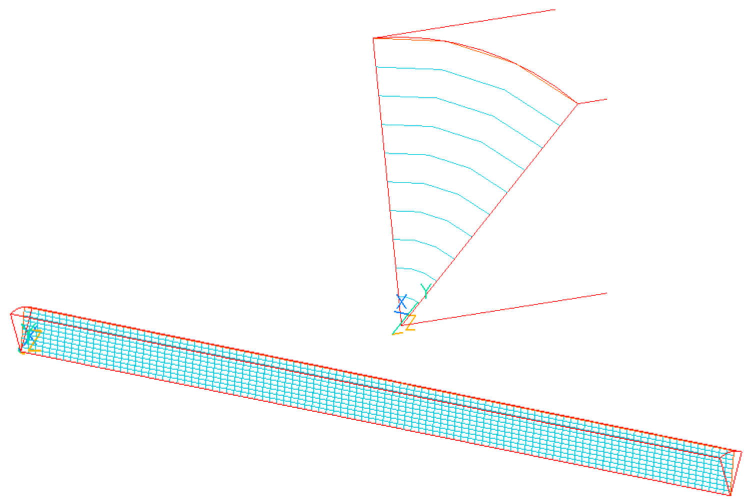
In the *Geometry* panel, uncheck *X-Auto, Y-Auto* and *Z-Auto* and set it to *–Manual.* Set the number of cells.

Grid Mesh Settings

Co-ordinate system	Time dependence			
Cylindrical-polar	Steady			
Inner radius	0.000000			m
Cut-cell method	SPARSOL	Settings		
	X-Manual	Y-Manual	Z-Manual	
Domain size	0.520000	0.075000	???	m
Domain origin	0.000000	0.000000	0.000000	m
Number of cells	1	10	100	
Tolerance	1.000E-3	1.000E-3	1.000E-3	m
No of regions	1	1	1	
Modify region	1	1	1	
Size	0.520000	0.075000	???	
Distribution	Power law	Power law	Power law	
Cell power	Set	Set	Set	
Cells in region	1	10	100	
Power/ratio	1.000000	1.000000	1.000000	
Symmetric	No	No	No	
Edit all regions in	X direction	Y direction	Z direction	

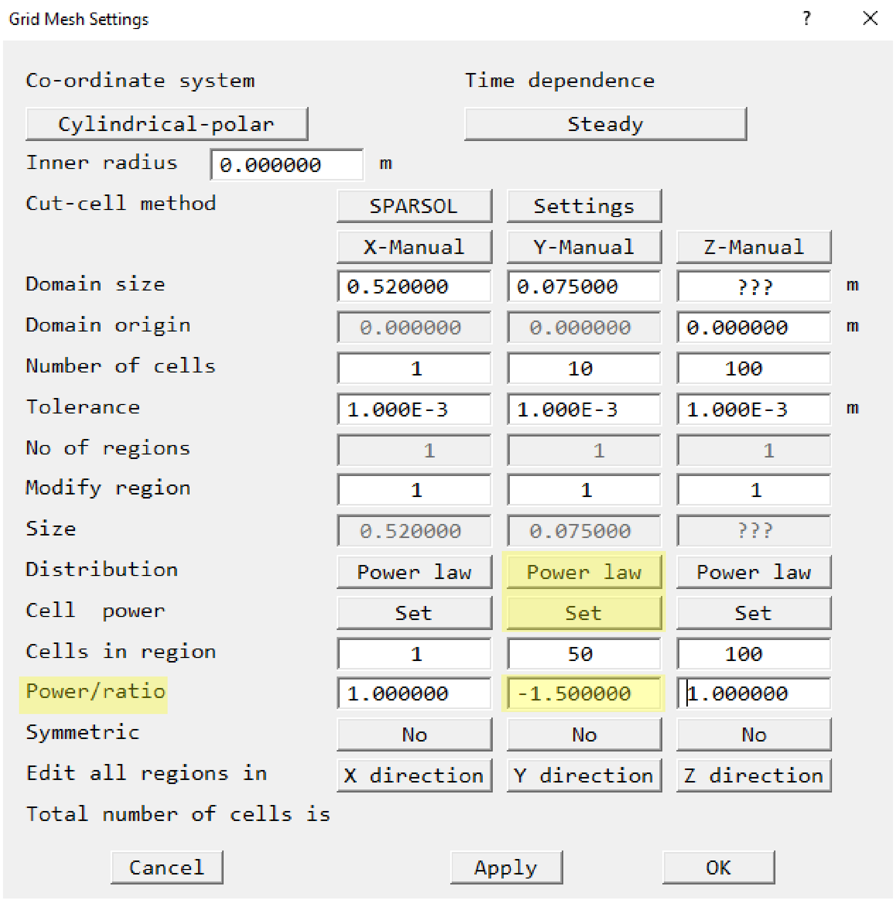
Total number of cells is 1000

Cancel Apply OK



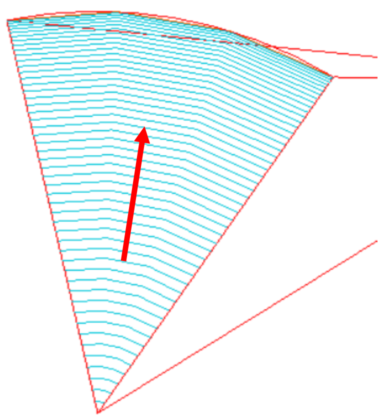
N.B. Axi-symmetry is automatically imposed by PHOENICS. No need to discretize in X.

Cells can be distributed non-homogeneously along a direction through a *power law*.



Grid Mesh Settings

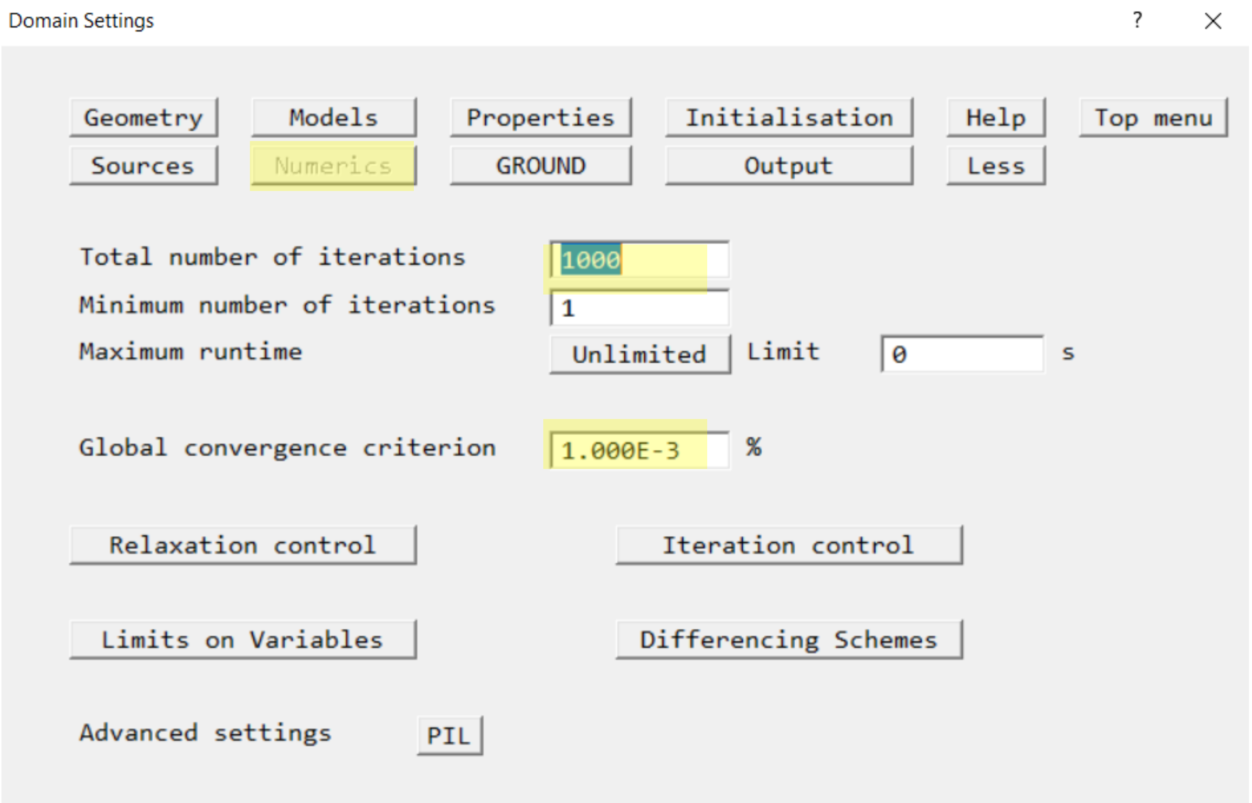
Co-ordinate system		Time dependence		
Cylindrical-polar		Steady		
Inner radius	0.000000 m			
Cut-cell method	SPARSOL	Settings		
	X-Manual	Y-Manual	Z-Manual	
Domain size	0.520000	0.075000	???	m
Domain origin	0.000000	0.000000	0.000000	m
Number of cells	1	10	100	
Tolerance	1.000E-3	1.000E-3	1.000E-3	m
No of regions	1	1	1	
Modify region	1	1	1	
Size	0.520000	0.075000	???	
Distribution	Power law	Power law	Power law	
Cell power	Set	Set	Set	
Cells in region	1	50	100	
Power/ratio	1.000000	-1.500000	1.000000	
Symmetric	No	No	No	
Edit all regions in	X direction	Y direction	Z direction	
Total number of cells is				
Cancel		Apply	OK	



Decreasing height along Y

Numerics
Number of iterations

From *settings* → *domain attributes* → *numerics*, set the maximum number of iterations and a global convergence criterion.



Note
Calculation of the shear stress

The shear stress profile in the fully developed region can be calculated through the following formula

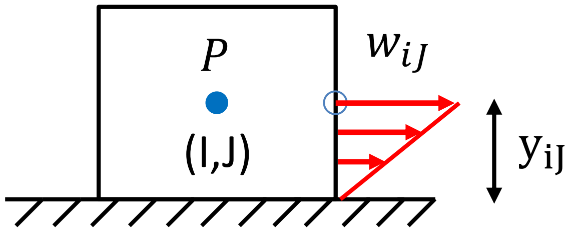
$$\tau_{rz}(r,\theta,z) = -2\mu\,\frac{1}{2}\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial r}\right) = -\mu\,\frac{dw}{dr}$$

The numerical calculation of the derivative dw/dr can be done either

1) by referring to the PHOENICS variable DWDY (note that, in PHOENICS, the variable DWDY is stored in the cell centres)
2) by approximating the derivative starting from the W1-distribution (pay attention to how the derivative is approximated and where the derivative is stored according to the selected scheme!)

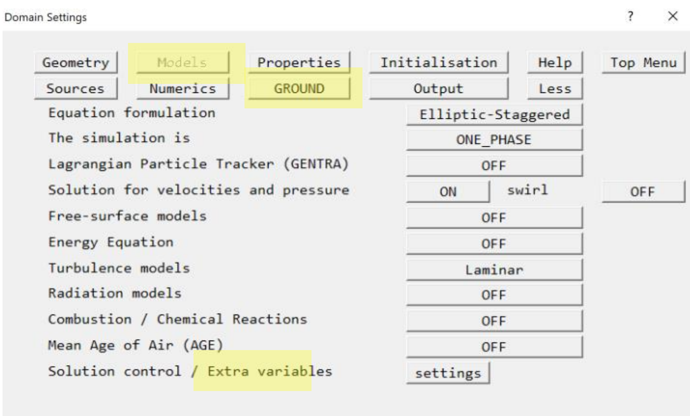
Note that, in laminar flow, the W1 is assumed linear between the wall and the first grid node, therefore

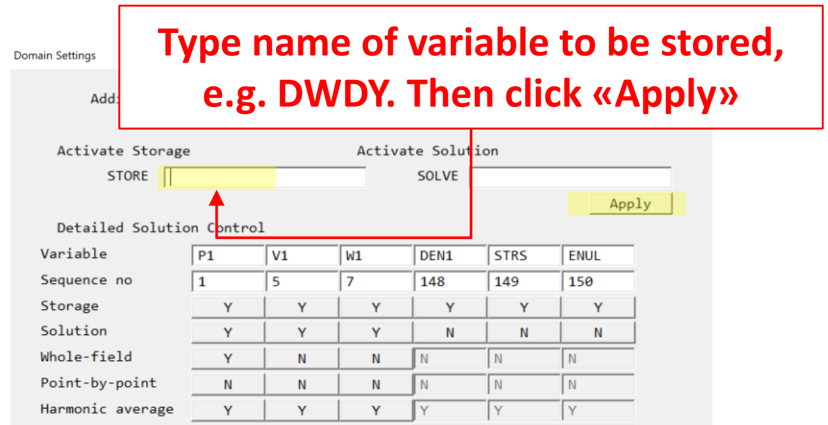
$$\tau_w^{CFD,lam} \approx \mu\,\frac{w_{iJ}}{y_{iJ}}$$



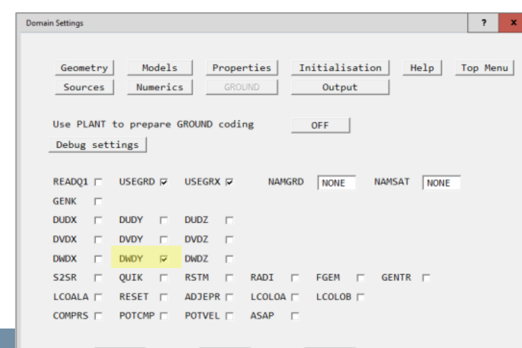
How to store additional variables

Users can store all variables in the following list
https://www.cham.co.uk/phoenics/d_polis/d_enc/reserv.htm
The list includes, for instance, the variables DWDY and DWDZ.
In order to store these variables in the .phi solution file, go to *Models > Solution control / Extra Variables*



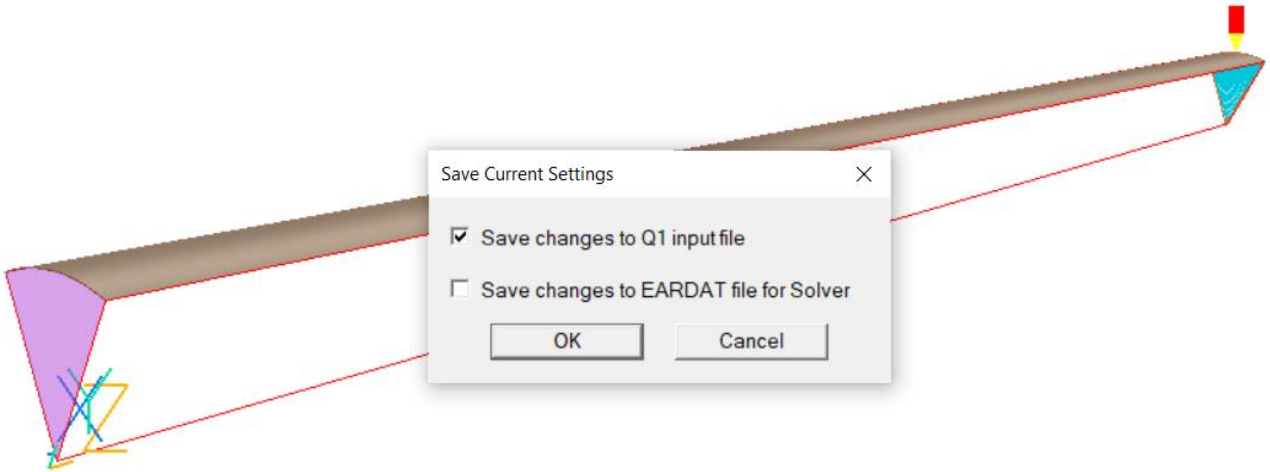


For derivatives, go to *GROUND* and flag the quantity



Save
Store q1 file

Go to *File → Save As a Case* and toggle on the *Save changes to Q1 input file*. Then select a proper folder for storing the «instruction» *q1* file.



Run
Convergence monitoring: EARTH solver

Go to *Run* → *Solver.* Click *Ok* on the pop-up windows. The EARTH solver will open.

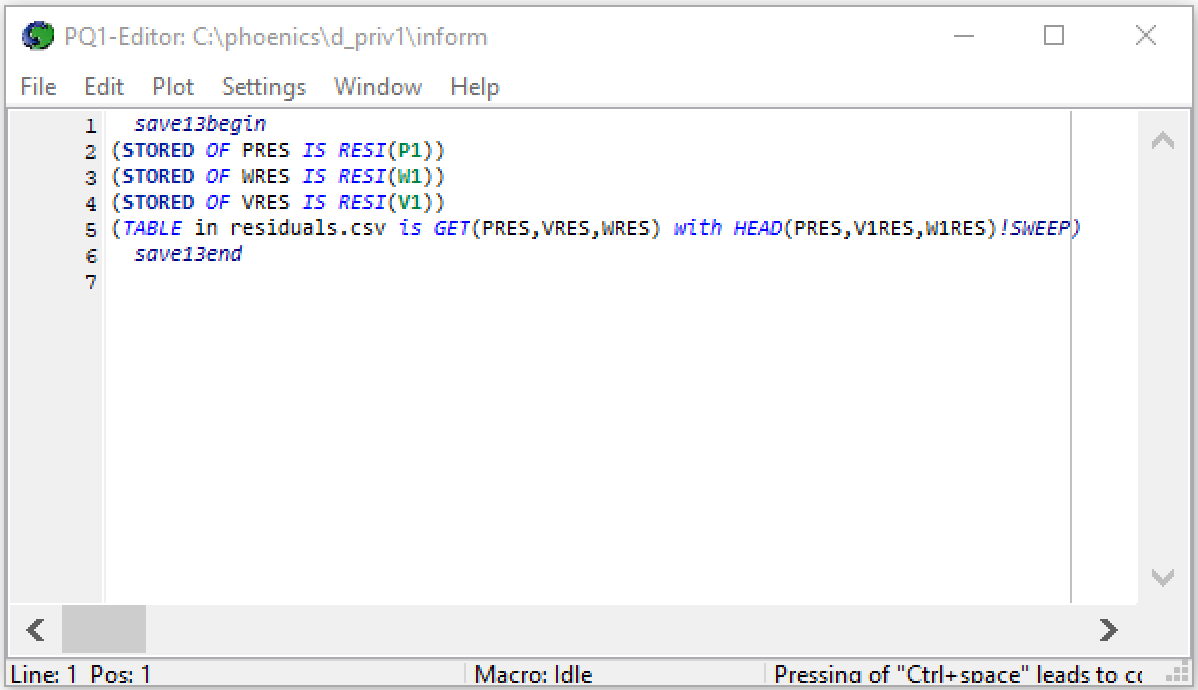
Values of the main output variables (P1, V1, W1) at probe position.

Whole field residual for the main control variables (P1, V1, W1)

Note: this is the Classic Solver GUI. Go to *Options* → *Solver Monitor Options* → *Monitor GUI Style = Classic*

Run (optional)
Convergence monitoring: residuals.csv

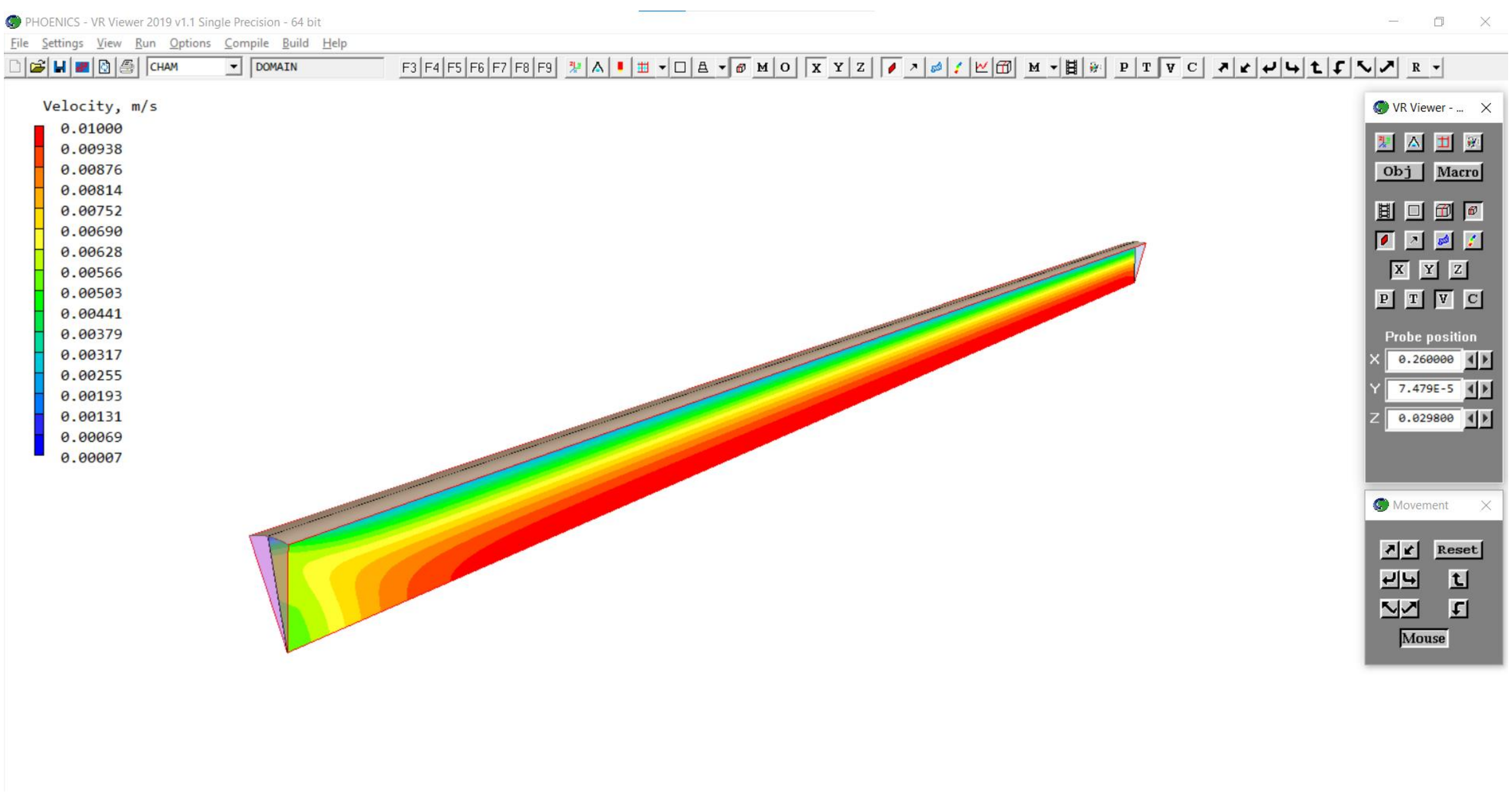
It is possible to export residuals to a .csv format through the InForm language. Go to *Settings* → *Domain Attributes* → *INFORM* → In *All Save blocks* select *save13* and press *Edit Save block*. Then insert the following code and *Save*.



A table named «residuals.csv» will be created in the *dpriv_1* folder. It will store the residuals for pressure and velocity at each sweep (or iteration).

Post-processing
VR-Viewer

Once the calculation has finished, go to *Run* → *Post Processor* → *GUI Post Processor (VR Viewer)* to preview the results. Select the result *phi* file to visualize the result.



Post-processing
Matlab import

In order to answer the questions provided in the *Requests*, it is recommended to use Matlab.

A script for importing formatted files from PHOENICS is provided (*XYZ_reduced_19_20.mat).* The following variables are imported:

- NX,NY, NZ – are the number of cells in X, Y and Z directions
- P1, V1, W1,… – are the matrices of the computed variables
- X_C, Y_C, Z_C – are the coordinates vectors of the cells' centers.
- X_E, Y_N, Z_H – are coordinates vectors of the cells' centers.